Louis J. Kiraly
Lewis Research Center
Cleveland, Ohio

(NASA-TM-101987) ADVANCES IN COMPUTATIONAL DESIGN AND ANALYSIS OF AIRBREATHING PROPULSION SYSTEMS (NASA. Lewis Research Center) 19 p CSCL 21E

N89-23465

Unclas
G3/07 0207404

NASA

# ADVANCES IN COMPUTATIONAL DESIGN AND ANALYSIS OF AIRBREATHING PROPULSION SYSTEMS

John M. Klineberg, Director
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135

E-4689

## Abstract

The development of new commercial and military aircraft depends, to a large extent, on engine manufacturers being able to achieve significant increases in propulsion capability through improved component aerodynamics, materials, and structures. The recent history of propulsion has been marked by efforts to develop computational techniques that can speed up the propulsion design process and produce superior designs. The availability of powerful supercomputers, such as the NASA Numerical Aerodynamic Simulator, and the potential for even higher performance offered by new parallel computer architectures, have opened the door to the use of multidimensional simulations to study complex physical phenomena in propulsion systems that have previously defied analysis or experimental observation. This paper provides an overview of several NASA Lewis Research Center efforts that are contributing toward the long-range goal of a "numerical test-cell" for the integrated, multidisciplinary design, analysis, and optimization of propulsion systems. Specific examples in internal computational fluid mechanics, computational structural mechanics, computational materials science, and high performance computing are cited and described in terms of current capabilities, technical challenges, and future research directions.

## Introduction

From the initial development of the airplane, the propulsion system has proven to be a pacing and limiting technology.[1] The development of new commercial and military aircraft has depended, to a large extent, on engine manufacturers being able to achieve significant increases in propulsion capability through improved component aerodynamics, materials, and structures. For the most part, these increases have been achieved in an evolutionary way with incremental progress based on the last successful design. Occasionally, such as in the development of ultra-quiet, highly efficient commercial aircraft and high-performance military aircraft, it has been necessary to make more dramatic changes in propulsion technology.[2] In these cases, government and industry researchers have worked together to devise and demonstrate new propulsion concepts, such as high speed turboprops and convertible rotorcraft engines. Today, we find ourselves facing a whole new set of propulsion challenges that will require revolutionary, rather than evolutionary, advances in propulsion technology. These include powerplants for supersonic cruise transports, vertical takeoff and landing aircraft, hypersonic, and transatmospheric vehicles.

A major consideration when embarking on the development of a new propulsion system has to be the enormous time and cost associated with taking a new concept through certification. As shown in Fig. 1, this process can take a decade with costs in the billions of dollars. Much of this time and cost can be attributed to the heavy dependence on hardware testing to accomplish system integration and to demonstrate performance and durability. A typical development program might require four or five major iterations in the design-build-test cycle.

The recent history of propulsion has been marked by efforts to develop computational techniques that can speed up the propulsion design process and produce superior designs. Advancements in such disciplines as computational fluid dynamics, computational structural mechanics, and computational materials science have produced a number of noteworthy successes. One example is the now-standard method of designing contours of propeller, compressor, and turbine airfoils using compressible flow and boundary layer calculations. Today, the availability of powerful supercomputers, such as the NASA Numerical Aerodynamic Simulator (NAS), has opened the door to the use of multidimensional simulations to study complex physical phenomena that have previously defied analysis or experimental observation.[3]

Still, much needs to be done to achieve "computation to flight" for propulsion systems. The complexity of propulsion systems and the environments in which they operate continue to present formidable challenges to those who attempt to build computer models that can credibly predict the aerodynamic, structural, and material behavior of those systems. Some of the major challenges are: (1) complicated geometries and boundary conditions, (2) three-dimensional, transient behavior, (3) chemical reactions, (4) heat transfer, (5) rotating systems, (6) material and structural modeling from the molecular level to the continuum, (7) aero-thermal-structural coupling, (8) interactions between propulsion components, and (9) interactions between the propulsion system and the airframe.

To achieve significant reductions in the time and cost of propulsion system development will require major advances in propulsion system modeling, algorithm design, and computational technology. Efforts are underway at NASA Lewis to develop many of these key, enabling technologies. These efforts are characterized by close coupling of algorithm and code development, by benchmark experiments to support modeling and code validation, and by the application of advanced, high performance computing technology to enhance both analyses and experiments.

This paper provides an overview of several NASA Lewis research efforts that are contributing toward the long-range goal of a "numerical test-cell" for the integrated, multidisciplinary design, analysis, and optimization of propulsion systems. Specific examples in internal computational fluid mechanics, computational structural mechanics, computational materials science, and high performance

computing are cited and described in terms of current capabilities, technical challenges, and future research directions.

## Internal Computational Fluid Mechanics

In the area of internal computational fluid mechanics (ICFM), research efforts at NASA Lewis are being directed toward the development of improved computational tools for both analysis and design.[4] The long-term goal of the NASA Lewis ICFM effort is to develop a capability to compute the aerothermodynamics of the complete propulsion system, from inlet to nozzle, in REASONABLE TIME, for REASONABLE COST, and most importantly, with KNOWN ACCURACY. In this context, the requirements for success are defined by the needs of the analyst, who must be able to use the computational tools on a routine basis. Currently, only independent pieces of the analysis system are available. Figure 2 signifies the range of codes that are currently being used or developed at NASA Lewis for aerothermodynamic analysis of propulsion components. The following paragraphs give specific examples of the use of these codes for analysis of inlets, turbomachinery, and combustors.

### Inlets

PEPSIS is a three-dimensional, viscous, parabolized Navier-Stokes (PNS) code that has been used in combination with wind tunnel experiments to support the development of inlets for hypersonic aircraft.[5] Figure 3 shows results from the analysis of a Mach 5.0 inlet configuration. Contours of pitot pressure are shown in Fig. 3(a) for various streamwise-normal planes in the supersonic portion of the inlet. A more detailed view of the last plane is shown in Fig. 3(b). The numerical simulation has pointed to the presence of strong secondary flows which would be detrimental to the performance of the inlet. Wind tunnel tests have confirmed the presence of the secondary flows. The PNS code has been used to redesign the inlet and a full-scale model of the redesigned inlet has undergone testing at NASA Lewis.

### Turbomachinery

The two-dimensional, steady, inviscid, blade passage codes, MERIDL and TSONIC, have been combined with a boundary layer analysis code to form a quasi-three-dimensional passage analysis system.[6] That system includes the computation of surface heat transfer as well as aerodynamic performance. Its usefulness has been demonstrated by Civinskas, et al. in assessing the effectiveness of a leading edge redesign in reducing peak heat transfer to a turbine blade.[7] The quasi-three-dimensional analysis was able to show that the desired decrease in heat transfer could be accomplished without compromising aerodynamic efficiency.

The capability to do unsteady, rotor-stator interaction calculations for turbomachines is being developed at NASA Lewis and NASA Ames. The required codes consume huge amounts of computer storage and CPU time due to the time-dependent nature of the calculations. Recently, Jorgenson and Chima reported on a technique (i.e., implicit residual smoothing) that results in a speedup of the calculations by a factor of about 4.5 without any effect on the accuracy of the solution.[8]

Three-dimensional flow simulations of multistage turbomachinery are being conducted at NASA Lewis using the method of Adamczyk.[9] In this method, an "average-passage equation system" is used to represent the time-averaged flow field within a typical passage of a blade row embedded within a multistage configuration. The model includes the effects of unsteadiness, compressibility, and viscosity. The M-STAGE code has been used to analyze a variety of turbomachinery configurations including the space shuttle main engine (SSME) turbopump.[10] Computed results have shown good qualitative agreement with experimental data.

Although the major emphasis of the NASA Lewis program in ICFM is on computational analysis, there has been a noteworthy accomplishment in the effort to develop improved numerical design tools. Sanz has developed an automated inverse design method that can compute a rotor or stator airfoil shape based on designer-specified flow parameters.[11] The inverse design method has been verified experimentally by Neal and Fleeter.[12] The authors built and tested a cascade of airfoils that had been designed by the Sanz code. Figure 4 shows comparisons between experimentally-determined and computed pressure distributions.[13,14] The agreement for this case was quite good. Currently, the automated inverse design code is being evaluated by the gas turbine industry.

### Combustors

A major advance in the use of CFD for design of high speed propulsion systems has occurred with the development of the RPLUS code.[15] RPLUS provides a new capability to perform time-efficient studies of mixing and chemical reactions in the flow fields of ramjets and scramjets. The three-dimensional version of the code, RPLUS3D combines the complete three-dimensional Navier-Stokes equations with species transport, an 8-species, 14-step, finite-rate chemistry model, and the Baldwin-Lomax turbulence model. An implicit, finite-volume scheme with lower-upper symmetric successive overrelaxation (LU-SSOR) is used to solve the fully-coupled equations. The RPLUS3D code has been tested in a series of calculations of a reacting hydrogen jet in crossflow. Fig. 5 shows water vapor contours resulting from the reaction of a sonic hydrogen jet in a Mach 4.0 crossflow. The RPLUS3D results compare favorably with previously reported results from a time-marching code while requiring much less computer time to converge to a steady solution.

Improvements in numerical algorithms, geometric modeling, grid generation, and parameter modeling, as well as dramatic improvements in supercomputer processing speed and memory, are now making ICFM a powerful tool for the aerodynamic analysis of propulsion systems. However, validation of existing and forthcoming models and codes is needed to gain confidence in the use of the codes as design tools. At NASA Lewis, code validation is being accomplished through close coupling of the code development efforts and "benchmark" experiments that focus on the basic flow mechanisms.

## Computational Structural Mechanics

From the "structures" viewpoint, aeropropulsion systems present many diverse and difficult simulation problems. To address these problems, the Structures Division of NASA Lewis is engaged in a broad research program which encompasses virtually all of the scientific disciplines involved in structural analysis and design. Research efforts are aimed at the development of theoretical models, computational methods, and experimental techniques. The focus of the discussion here is computational methods and two specific examples are described that represent the current capability and future directions of research in this area.

### Structural Tailoring

A recent accomplishment which exemplifies current capability in computational structural mechanics for aeropropulsion is the development of an improved methodology for engine blade design. The concept of structural tailoring has been successfully demonstrated in applications to cooled turbine blades and advanced propfan blades. In the latter effort, the structural tailoring of advanced turboprops (STAT) design system was developed to provide a superior alternative to traditional propeller design practice.[16]

The traditional design procedures combine engineering and art. The process requires a team of experienced design engineers who can make the trade-offs that are necessary to satisfy aerodynamic and structural performance requirements. Not surprisingly, this process tends to be time-consuming (expensive), cumbersome (error-prone), and subjective (influenced by designer judgment). As such, it is difficult to arrive at a satisfactory design, much less an optimum one. The STAT design system streamlines, automates, and formalizes the process by integrating discipline-specific analyses and mathematical optimization into a computationally effective package.

The STAT design system adjusts internal (composite construction) and external (dimensions/thickness ratios, sweep, twist, etc.) geometry to achieve an objective of either: (1) minimum aircraft direct operating cost for a full-size blade, or (2) minimum aeroelastic differences between full-size and scaled (wind tunnel model) representations. To determine the objective function parameters and to enforce design constraints, STAT incorporates discipline-specific analyses to evaluate aerodynamic efficiency, stresses and frequencies, forced response, flutter stability, and acoustic characteristics.

Several novel strategies are employed in STAT to improve computational effectiveness. Approximate (more efficient) analyses are used during the optimization process and then higher fidelity (but compute-intensive) analyses are used to verify the optimum design. To reduce the likelihood of producing an infeasible design, recalibration of the analysis is accomplished by "tuning" the approximate analyses using results from the higher fidelity analyses. Related developments are underway to enhance the STAT system by employing techniques that will accelerate the higher fidelity calculations. One effort involves adaptation of the multigrid strategy, which has been used successfully for finite-difference computations in fluid dynamics, to the finite-element computations, more commonly used in structural dynamics.[17]

STAT has demonstrated its superiority over the traditional design process for propfan design. Not only has STAT produced better designs, it has resulted in dramatic savings in design effort, as well. For the example shown in Fig. 6, the traditional process required over 100 manual design iterations, many man-months of design effort, and substantial computer resources. The STAT process required a few manhours and about 2 hr of computer time. These statistics provide clear evidence of the STAT design system's potential for reducing design cycle time and cost.

### Probabilistic Structural Mechanics

A new and growing research area in Computational Structural Mechanics is probabilistic structural mechanics. Successes in this area hold significant promise for reducing the enormous burden of hardware certification testing that now dominates the development cycle for new propulsion systems.

The fundamental variables in a structural design (i.e., geometry, material, boundary conditions and loads) are, in reality, uncertain quantities. Even though this uncertainty contributes to the risk of component/system failure, the degree of risk is generally not assessed; the reason being that traditional "deterministic" design methodology does not provide a means to accomplish such an assessment. In an attempt to avoid the acknowledged risk, deterministic methodology makes use of the "safety factor" in design and extensive "proof" testing of hardware. This approach to reducing risk is expensive, both in cost of development and in cost of operating at reduced performance efficiency. This approach may be unacceptable in situations where operating economy and reliability are the principal (and competing) design objectives.

In the interest of maximizing safety and minimizing cost, efforts have begun to formally quantify uncertainty and attendant risk. Early accomplishments have provided a framework within which computational methods are being developed for probabilistic structural analysis and reliability-based design. These efforts have produced the numerical evaluation of stochastic structures under stress (NESSUS) analysis system.[18] NESSUS features a modular software structure comprising: (1) an expert system that serves as a user interface, (2) modules for performing finite-element and boundary-element perturbation analyses, and (3) probability simulation and integration modules. Essentially, NESSUS determines the cumulative probability of occurrence of structural response variables and quantifies structural reliability for defined failure events (Fig. 7). NESSUS provides a capability to evaluate cost versus safety tradeoffs in a structural design and to develop risk acceptance criteria.

Particularly noteworthy are two numerical "breakthroughs" that have made probabilistic analysis of realistic structures computationally tractable. The first is the development of efficient iterative perturbation algorithms for computing the sensitivities of response variables to small fluctuations of random independent variables. These algorithms have been adapted to the more modern

(and elaborate) mixed-field, finite element formulations.[19] The second development is the fast probability integration (FPI) algorithm which translates the sensitivities into a cumulative probability of occurrence.[20] The FPI algorithm has provided an efficient alternative to the well-established Monte Carlo simulation techniques.

Near-term efforts in probabilistic analysis will extend the applicability of NESSUS to nonlinear regimes of structural response. Longer-term efforts are envisioned that will combine the probabilistic analysis methodology with damage state and failure models, producing a comprehensive capability for assessing structural reliability.

## Computational Materials Science

Materials research at NASA Lewis encompasses a large field of study with applications as varied as structures, semiconductors, solar-cells, high-temperature engine components, lubricants, and coating technology. This work involves many classes of materials, including metals, intermetallics, ceramics, polymers and semiconductors. Of special interest are composites with metal, ceramic, or polymer matrices.

A central thrust in the materials program at NASA Lewis is the development of high-temperature materials for jet engine applications. Over the past several decades, NASA Lewis researchers, in conjunction with many academic and industrial partners, have developed several new materials that have contributed significantly to the present state of jet engine technology.

While there have been dramatic advances in the PRACTICE of materials science, computational approaches are still in their infancy. The science of materials is still largely an experimental art. This is not the result of a lack of appreciation for the potential of computational materials science, but is caused by the inherent difficulties associated with computing materials phenomena that simultaneously range in scale from the continuum regime down to the molecular level.

An examination of a typical materials science process will usually show the coexistence of solid, liquid, and gaseous phases, and of numerous chemical species which can interact to form still other species. The classical continuum boundary conditions are supplemented by the presence of several unique conditions determined by the local thermodynamic state. An additional complication is the formation of shifting interfaces among the different phases. Computationally, such problems are extremely difficult to solve, and until recently, no efficient algorithms were available for general three-dimensional, time-dependent simulations.

Simulation at the molecular level requires a completely different viewpoint. Here the main interest is in the way individual atoms and molecules attach to form complex new structures, such as polymers and crystals. Computational approaches involve tools from statistical mechanics, molecular modeling and other specialized techniques.

The hallmark of the difficulties in computational materials science is that the two levels of approximations, vastly different both in scale and in governing physics, are intimately intertwined in the description of a single process. At NASA Lewis, a newly developed thrust in computational materials science is addressing both the molecular and the continuum regimes. The following paragraphs discuss one example of that work, namely, the chemical vapor deposition (CVD) of silicon for high temperature applications.

### Chemical Vapor Deposition

The CVD process consists of injecting a nutrient gas (silane) with a carrier gas mixture (hydrogen with argon or helium) into a reactor. The reactor may take many shapes, but principally it is a container with water or gas-cooled walls. The sample (susceptor) to be coated is placed inside the reactor, and is heated (commonly by an external RF field). As the initially cool gas stream passes over the hot sample, several simultaneous chemical reactions occur, producing additional chemical species. When species arrive at the heated sample, surface reactions follow, whose final result is the deposition of silicon over the sample. Such deposits, while of interest in their own right, are precursors to silicon carbide coatings and bulk silicon carbide engine components.

The approach being taken at NASA Lewis, represents the state-of-the-art in the modeling of the CVD process. Three-dimensional fluid, heat, and mass transfer are considered, together with homogeneous (gas stream) and heterogeneous (surface) finite chemical reaction rates, the effects of buoyancy, and the local thermodynamics of the system. Studies are underway to test and validate the code and to use the code to determine the relative importance of different carrier gases in affecting the final spatial distribution of the silicon deposits on the sample's surface. The goal is to develop a simulation that can be used by designers to determine the optimum set of geometry and operating parameters for achieving desired deposition characteristics.

The reactor represented in Fig. 8(a) is an experimental apparatus with which silicon deposition rates have been measured for many combinations of boundary conditions and inlet parameters. The results represent an excellent data base for the validation of the numerical approach.

Figures 8(b) and (c) show flow visualization results for simulation runs using a 75/25 mixture of $H_2$ and Ar for the carrier gas. Results are shown for simulations of both earthbound (1 g) and spaceflight (nominally 0 g) conditions. The simulation shows the dramatic effects of buoyancy in producing a "cork-screw-effect." The simulation results indicate that the final deposition rates and spatial distribution on the susceptor will be nonuniform and highly complex for carrier gases of $H_2$ and Ar in the 1 g environment. Surprisingly, similar simulations with mixtures of $H_2$ and He showed only minor differences between the 1 g and 0 g cases. The simulation has provided the insight that a simple replacement of Ar with He could suppress the major adverse effects of natural convection. It should also be pointed out that simple, dimensional analysis, using natural convection parameters such as the Grashof and Rayleigh numbers, could not be expected to provide similar insights into the flow regimes expected. This is mainly because of the steep thermal gradients

present (~700 K/cm) and the three-dimensional nature of the flow.

Figure 9 shows computed concentration fields of silane in three transverse slices along the susceptor for the $H_2$/Ar case. The three-dimensional aspect of the natural convection in the 1 g case is apparent. The concentration field is nonuniform in the cross-flow direction with entrainment of high levels of unused silane in the cork-screw vortex, as well as above the susceptor, where a smaller vortex is found. The fields for the 0 g case show diffusion-controlled transport, with the expected downstream depletion of the nutrient gas (silane) due to the simultaneous reactions and the depositing of silicon on the susceptor. When the flow is diffusion controlled, even two-dimensional simulations can provide good results. Two-dimensional models and the simpler boundary-layer analyses are currently the common means to simulate CVD processes. However, as the NASA Lewis results have shown, the use of the simpler models must be based on the results of a full, three-dimensional simulation. To initially assume that the three-dimensional effects can be neglected is dangerous and may result in substantial errors.

Purely experimental approaches to research in material science can no longer guarantee the development of new and improved materials for high-temperature applications. Since a suitable test matrix may involve hundreds of experiments required to optimize a process, the attractiveness of the computational approach is evident. Coupled with the exploitation of the latest advances in algorithms and hardware, computational materials science has the potential to provide needed guidance in a timely manner and a true working partnership with experimental methods.

## High Performance Computing

The preceding sections have illustrated current capabilities, technical challenges, and research directions in internal computational fluid mechanics, computational structural mechanics, and computational materials science. As indicated, computational capabilities are enabling, and in some cases, pacing many of the critical advances in these disciplines. In this section, we consider some of these computational requirements in both qualitative and quantitative terms and present the NASA Lewis perspective on the key issues in high performance computing that impact those requirements.

### Computational Requirements

Internal fluid mechanics. Experience with the PROTEUS code has provided some measure of the computational requirements associated with time-accurate simulation of inlets, ducts, and nozzles. The PROTEUS code has shown a scalable CPU time requirement of $3.6 \times 10^{-4}$ sec/grid point/time step on the Cray X-MP. A realistic, two-dimensional model often requires about 10 000 time steps on a 100 by 100 grid of meshpoints to ensure convergence of the solution. This yields a time to solution of about 10 CPU hours on a Cray X-MP. Extrapolating these results to hypersonic inlet analyses that would require a three-dimensional representation with a grid size of 100 by 100 by 100 yields CPU times of about 1000 CPU hours. Clearly, dramatic advances in computing technology are needed to enable these calculations to be used on a regular basis for engineering applications.

SSME turbopump calculations, using Adamczyk's M-STAGE, have required a mesh size of 110 by 11 by 11 to discretize the geometry. The code required approximately 800 CPU seconds to develop adequate initial conditions for the iterative solution and 4000 sec for convergence in a typical case. Use of the code to investigate localized phenomena, such as blade tip vortex generation and propagation, would require much finer grids. Some initial work along this line has required mesh geometries of 200 by 40 by 21 and CPU times from 50 to 200 hr/case on the Cray X-MP. In a design process, many such cases would have to be run to develop performance profiles that can guide the selection of the "best" design parameters.

Structural mechanics. One of the critical issues in the development of advanced propulsion systems and one that poses significant computational challenges is the prediction of component life. Researchers at NASA Lewis have been developing and evaluating unified, constitutive models for life prediction.[21] Most of the analyses use the MARC or NASTRAN finite element codes in conjunction with a number of NASA Lewis-developed codes.[22] Typically, a thermal/structural code, such as MARC, is used to determine the structural and thermal loading distribution. These loadings are input to stress-strain life models to identify the critical point(s) of the structure. The process may have to be repeated several times to evaluate alternative geometries and/or materials. Any changes that might impact the flow or heat transfer characteristics of the component have to be reanalyzed by the fluid dynamicist. This type of analysis has been used to study the cowl lip of a hypersonic inlet. There, a complete analysis required 20 to 30 runs of the MARC code with each run typically taking about 5 or 6 CPU hours on the Cray X-MP.

### Computational Technology

Computational environment. The development and practical application of advanced numerical simulation codes for propulsion design and analysis will require significant increases in computing power (i.e., speed and memory). These advances will have to be matched by similar improvements in the major areas of computational support, namely, graphics, program development, and data management. The graphics tools are especially important because of the massive amounts of multidimensional data that need to be assimilated. The program development environment must allow noncomputer specialists to work effectively in programming, debugging, and running simulation codes on state-of-the-art, high performance computers. The retrieval, manipulation, and storage of extremely large programs and databases are critical throughout the simulation process.

NASA Lewis is moving as rapidly as possible toward the establishment of a high performance computing environment that will satisfy the projected needs in propulsion simulation. The approach being taken is to provide a local high performance graphics workstation capability with networked access to UNIX-based parallel processors, mainframes and supercomputers. NASA Lewis currently has about 50 such workstations, a broad-band and Ethernet-based cable network, Cray X-MP supercomputer, an Alliant

FX80 parallel processor, and an Amdahl 580 mainframe computer. NASA Lewis has just completed a migration from the proprietary Cray Operating System (COS) to the UNIX-based CRAY Operating System (UNICOS).[23] Detailed discussions of the utilization of UNICOS and the solid state storage device (SSD) for large programs and databases are given by Hackenberg.[24]

Parallel processing. As we approach the physical limits of electronic switching speeds, the potential for achieving significant increases in processor speed is diminishing. The NASA Lewis program in parallel processing is growing with the recognition that parallel processing is a viable approach to achieving the orders of magnitude increases in computing speed that will be needed for propulsion simulation. Currently, the NASA Lewis program includes grant support for and cooperative research programs with a number of universities, on-going evaluations of various commercially available systems, and cooperative arrangements to evaluate hardware and software under development by vendors.

The selection of a parallel processing architecture for a particular simulation problem is seldom clear-cut and depends on the nature of the parallelism inherent in the problem formulation. Figure 10 illustrates how the physical parallelism that exists in a multistage turbomachine was exploited by Mulac in mapping the M-STAGE code onto the four-processor CRAY X-MP/48.[10] In this case, the flow field calculations for each blade row of a two-stage turbine were assigned to separate processors. This approach to parallel processing is well suited for today's supercomputers that employ a small number of very powerful processors. Massively parallel processing systems, featuring hundreds or even thousands of less powerful processors, are beginning to emerge from research laboratories. These systems hold the promise of being able to exploit much finer-grain parallelism in the algorithms to deliver otherwise unattainable computing speeds. This concept is illustrated in Fig. 11 for the multistage turbomachinery problem. Here, clusters of processors might be organized in a hierarchical fashion to perform concurrent calculations at the task (blade row), sub-task (flow passage), and sub-sub-task (mesh point(s)) levels.

This movement toward parallel processing for computationally-intensive, time-critical applications, is being impeded by a continuing software gap with very few tools available for programming and operating parallel processing systems. These problems are compounded by the heavy dependence on Fortran as a programming language in the scientific and engineering disciplines. The basic programming constructs in Fortran are sequential in nature and serve to disguise whatever parallelism may exist in the problem. Some of the new languages, such as ADA, have features that restore some of the expressiveness and program control, but are generally not available on the new parallel processing machines.

With the proliferation of new hardware architectures and language implementations, it is unclear as to which systems will ultimately emerge as superior for propulsion simulation. Currently, NASA Lewis is pursuing two efforts, as illustrated in Fig. 12. The first addresses the Center's "production" workload and involves the acquisition and use of "conventional" supercomputers whose software will allow the programmer to use compiler directives in the Fortran code to accomplish a moderate degree of parallel processing. The second effort addresses the rapidly growing demands for high performance computing and involves the investigation of highly massively parallel architectures, algorithms, programming languages, etc. that hold the promise of being able to satisfy the computational workload in the years to come. The recent acquisition by NASA Lewis of an Alliant FX-80 system with eight processors can be viewed as a point of divergence between the two paths with future acquisitions aimed at providing access to state-of-the-art, massively parallel architectures.

NASA Lewis has initiated a number of pilot activities in computational technology to gain insight into many of the parallel processing issues. These activities include the development of a finite element solver that utilizes multiple transputer boards and is programmed using the OCCAM language. Also, NASA Lewis researchers have developed the multiarchitecture Hypercluster concept that is being used to investigate the matching of algorithms and architectures for ICFM applications.[25] That work is building on experience gained in the development and application of parallel processors for real-time jet engine simulation.[26]

## Putting It All Together - Numerical Propulsion System Simulation

The preceding paragraphs have illustrated how advances in the physical and computational sciences are enabling researchers to better understand, model, and simulate the physical processes that govern propulsion system performance, durability and life. While progress continues to be made in each of the propulsion disciplines, the overall impact of numerical simulation on propulsion system design and development remains limited, as evidenced by the long, costly, hardware-oriented engine development cycle. Major advances are needed to bring numerical simulation technology to the point where many of the critical design issues can be settled on the computer.

NASA Lewis is in a unique position to "put it all together" - to develop, demonstrate, and validate new simulation hardware and software procedures for multidisciplinary design, analysis, and optimization. To that end, many of the NASA Lewis organizations and research teams have begun to direct their activities toward a common, long-range goal or "vision," namely the development of technologies for a "numerical test-cell." The numerical test-cell, or numerical propulsion system simulation (NPSS), is illustrated in Fig. 13.

The NPSS concept hinges on our being able to achieve, exploit, and combine technical advances in physical modeling, algorithm development, and high performance computing. By incorporating the best available hardware and software emerging from disciplinary and multidisciplinary research programs, we will be establishing a powerful, and very flexible, simulation test-bed that represents the state-of-the-art in technology for multidisciplinary design, analysis, and optimization.

In discussing the implementation of NPSS, one encounters a recurring theme. That is, the need for increased integration - of people, ideas, and

tools. For the most part, numerical simulation efforts in propulsion have focused on single-discipline models of individual components. NPSS developers, while dependent on continued progress in the individual disciplines, must find a way to integrate those efforts. Many of the barrier problems in propulsion are multidisciplinary. The aeroelasticity of blades in advanced turboprops is an example. To accurately compute the onset of flutter or the nature of the forced vibration of blades, a numerical analyst must, at once, deal with both the aerodynamic and structural responses and their interactions.

At the present time, several NASA Lewis organizations have begun to cooperate on small, pilot projects that are dealing with some of the integration issues. Presently, there is a general lack of formalized procedures for developing multidisciplinary simulations of propulsion components and systems. Researchers in the Structures Division have begun to address this problem in the design of a computational engine structures simulator (CESS). The CESS provides an overall framework and software strategy for interfacing aero and structural modeling and analysis modules. This "loosely coupled" approach to multidisciplinary analysis is viewed as a first step in the NPSS design process that will eventually lead to coupling of the aero and structural algorithms and a unified, multidisciplinary code.

The need for increased computing power, discussed earlier for the individual disciplines, becomes even more acute when one considers the practical use of multidisciplinary codes for propulsion system design. "Hypercomputers" that are 100 to 1000 times faster than today's supercomputers will be needed to run coupled aero-thermal-structural simulations of propulsion systems.

While the challenges are great, the potential rewards warrant a concerted, long-term effort. Savings in time and cost of engine development, that would result from NPSS technology, have been estimated at between 25 and 40 percent.

It is expected that the NPSS effort will continue to grow with industry and academia playing key roles to ensure a high quality research and technology program that is responsive to the needs of the propulsion community.

## Acknowledgements

The work reported in this paper resulted from the efforts of researchers in the Internal Fluid Mechanics Division, the Structures Division, the Materials Division, and the Computer Services Division of the NASA Lewis Research Center. The author wishes to specifically acknowledge the contributions of Dr. Raymond Gaugler, Mr. Dale Hopkins, Dr. Arnon Chait, and Dr. Steven Sidik, who provided assessments of current capabilities and future needs in computational fluid mechanics, computational structural mechanics, computational materials science, and high performance computing, respectively. The author also wishes to thank Mr. John Szuch for contributing the material on NPSS and for his leadership in organizing and preparing the paper.

## References

1. "Aeronautics Technology Possibilities for 2000. Report of a Workshop," National Research Council, NASA CR-174018, 1984.

2. "Current Capabilities and Future Directions in a Computational Fluid Dynamics," National Research Council, NASA CR-179946, 1986.

3. Kutler, P. and Yee, H., eds., Supercomputing in Aerospace, NASA CP-2454, NASA, Washington, D.C., 1987.

4. Miller, B.A., Anderson, B.H., and Szuch, J.R., "Internal Fluid Mechanics Research on Supercomputers for Aerospace Propulsion Systems," NASA TM-100289, 1988.

5. Benson, T.J., "Three-Dimensional Viscous Calculation of Flow in a Mach 5.0 Hypersonic Inlet," AIAA Paper 86-1461, June 1986.

6. Boyle, R.J., Haas, J.E., and Katsanis, T., "Comparison Between Measured Turbine Stage Performance and the Predicted Performance Using Quasi-3D Flow and Boundary Layer Analyses," AIAA Paper 84-1299, June 1984 (NASA TM-83640).

7. Civinskas, K.C., Boyle, R.J., and McConnaughey, H.V., "Impact of ETO Propellants on the Aerothermodynamic Analyses of Propulsion Components," AIAA Paper 88-3091, July 1988 (NASA TM-101303).

8. Jorgenson, P.C.E. and Chima, R.V., "An Unconditionally Stable Runge-Kutta Method for Unsteady Flows," AIAA Paper 89-0205, Jan. 1989 (NASA TM-101347).

9. Adamczyk, J.J., Celestina, M.L., Beach, T.A., and Barnett, M., "Simulation of 3-D Viscous Flow Within a Multi-Stage Turbine," NASA TM-101376, 1989.

10. Mulac, R.A., Celestina, M.L., Adamczyk, J.J., Misegades, K.P., and Dawson, J.M., "The Utilization of Parallel Processing in Solving the Inviscid Form of the Average Passage Equation System for Multi-Stage Turbomachinery," 8th Computational Fluid Dynamics Conference, AIAA, New York, 1987, pp. 70-80.

11. Sanz, J.M., "Automated Design of Controlled Diffusion Blades," Journal of Turbomachinery, Vol. 110, No. 4, Oct. 1988, pp. 540-544.

12. Neal, J.H. and Fleeter, S., "Loading Effects on the Aerothermodynamics of an Annular Cascade of Three-Dimensional Airfoils," NASA Grant NSG-3285, Purdue Univ., West Lafayette, IN, Rept. ME-TSPC-TR-88-12, Mar. 1988.

13. Denton, J.D., "An Improved Time Marching Method for Turbomachinery Flow Calculation," ASME Paper 82-GT-239, Apr. 1982.

14. Chiang, H.W.D. and Fleeter, S., "Locally Analytical Prediction of the Steady Inviscid Incompressible Flow Through an Airfoil Cascade," International Journal of Computers and Mathematics with Applications, accepted for publication.

15. Shuen, J.S. and Yoon, S., "Numerical Study of Chemically Reacting Flows Using an LU Scheme," AIAA Paper 88-0436, Jan. 1988.

16. Brown, K.W., "Structural Tailoring of Advanced Turboprops (STAT)" PWA-5967-46, Pratt and Whitney Aircraft, East Hartford, CT, 1988, NASA CR-180861.

17. Kascak, A.F., "Multigrid for Structures Analysis," Lewis Structures Technology-1988, Vol. 1 - Structural Dynamics, NASA CP-3003-VOL-1, NASA, Washington, D.C., 1988, pp. 67-90.

18. Cruse, T.A., Wu, Y.T., Dias, J.B., and Rajagopal, K.R., "Probabilistic Structural Analysis Methods and Applications," Computers and Structures, Vol. 30, No. 1-2, 1988, pp. 163-170.

19. Dias, J.B., Nagtegaal, J.C., and Nakazawa, S., "Iterative Perturbation Algorithms in Probabilistic Finite Element Analysis," to appear in: Computational Mechanics of Probabilistic and Reliability Analysis, W.K. Liu and T. Belytschko, eds., Elme Press International, Lausanne, Switzerland, 1989.

20. Wu, Y.T. and Wirsching, P.H., "A New Algorithm for Structural Reliability Estimation," Journal of Engineering Mechanics, Vol. 113, No. 9, Sept. 1987, pp. 1319-1336.

21. Kaufman, A., Tong, M., Saltsman, J.F., and Halford, G.R., "Structural Analysis of Turbine Blades Using Unified Constitutive Models," NASA TM-88807, 1987.

22. Hopkins, D.A. and Chamis, C.C., "A Unique Set of Micromechanics Equations for High-Temperature Metal Matrix Composites," NASA TM-87154, 1985.

23. Pischel, K.A., "Experiences of a CDS to UNICOS User Conversion," Proceedings of 21st Semi-Annual Cray User Group Meeting, April 1988, pp. 293-297.

24. Hackenberg, T., "Configuring and Using a SSD Under UNICOS in Guest Mode," Proceedings of 22nd Semi-Annual Cray User Group Meeting, Sept. 1988, pp. 94-102.

25. Blech, R.A., "The Hypercluster - A Parallel Processing Test-Bed Architecture for Computational Mechanics Applications," NASA TM-89823, 1987.

26. Milner, E.J. and Arpasi, D.J., "Simulating a Small Turboshaft Engine in a Real-Time Multiprocessor Simulator (RTMPS) Environment," NASA TM-87216, 1986.

ORIGINAL PAGE IS
OF POOR QUALITY

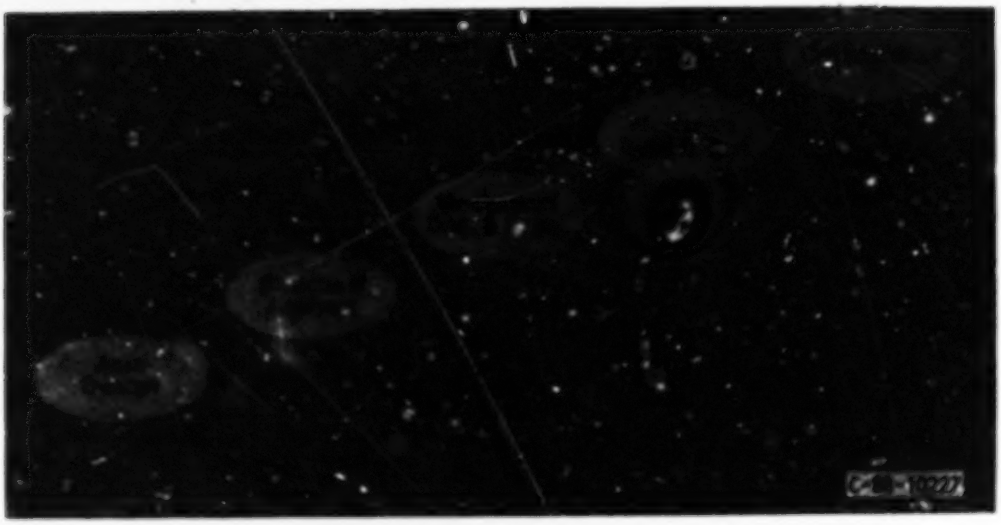
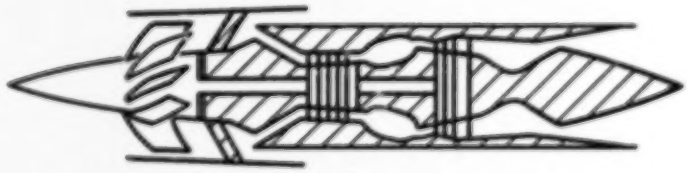


FIGURE 1. - AIRCRAFT ENGINE DEVELOPMENT CYCLE, TODAY'S METHODOLOGY.

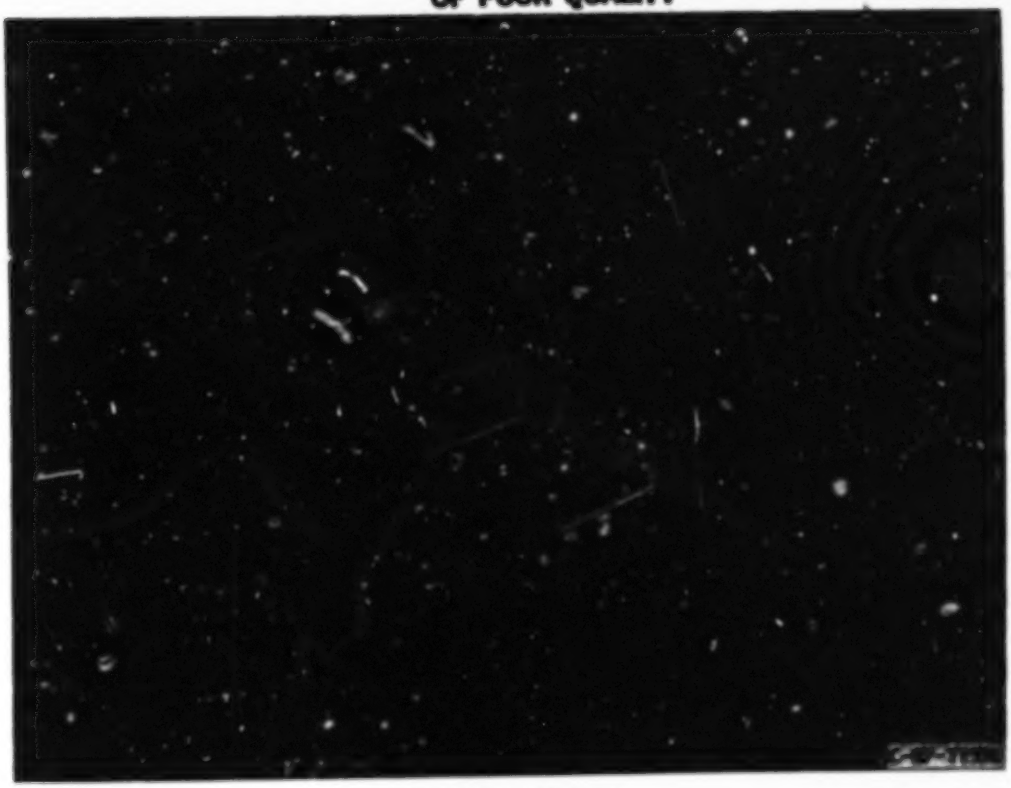


| FLOW REGIME \ COMPONENT | | | INLET | FAN | COMPRESSOR | COMBUSTOR | TURBINE | NOZZLE |
|---|---|---|---|---|---|---|---|---|
| 1D | STEADY | | ● LUMPED - PARAMETER, THROUGH-FLOW ENGINE MODELS | | | | | |
| | UNSTEADY | | | | | | | |
| 2D | STEADY | INVISCID | | ● | ● | | ● | |
| | | VISCOUS | ● | ● | ● | ● | ● | ● |
| | UNSTEADY | INVISCID | | | | | | |
| | | VISCOUS | | ● | ● | | ● | |
| 3D | STEADY | INVISCID | | ● | ● | | ● | |
| | | VISCOUS | ● | ● | ● | ● | ● | ● |
| | UNSTEADY | INVISCID | | | | | | |
| | | VISCOUS | | ● | ● | | ● | |

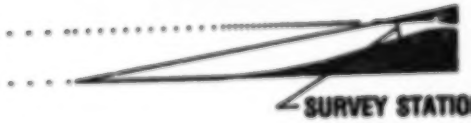
● EXISTING CODES

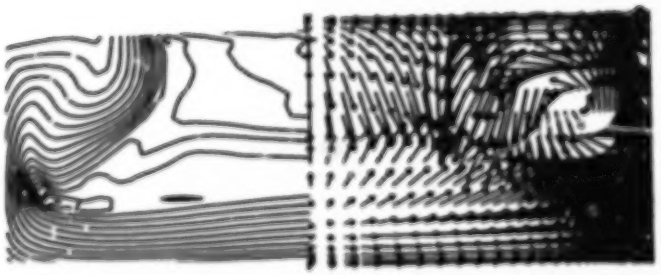
FIGURE 2. - AEROTHERMODYNAMIC ANALYSIS CODES IN USE AT NASA LEWIS.

ORIGINAL PAGE IS
OF POOR QUALITY



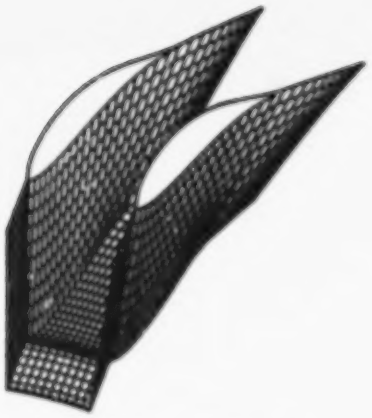
(A) PRESSURE CONTOURS.



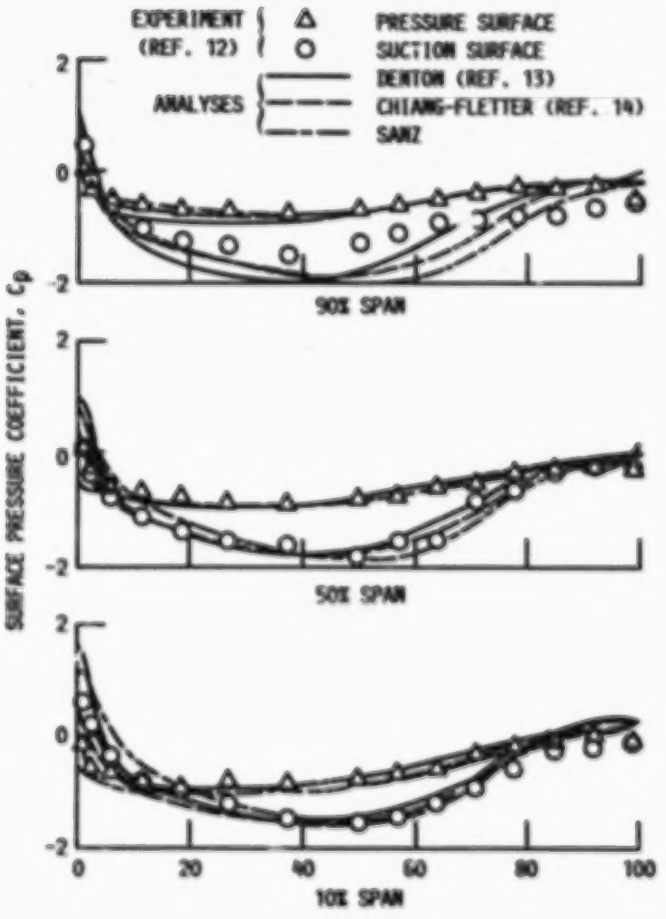




(B) MACH NUMBER AND SECONDARY VELOCITY VECTORS.

FIGURE 5. - MACH 5.0 INLET ANALYSIS.



(A) AIRFOIL GEOMETRY.




(B) PRESSURE COEFFICIENT.

FIGURE 4. - COMPARISON OF PREDICTED AND MEASURED PERFORMANCE OF AIRFOILS DESIGNED BY INVERSE METHOD.

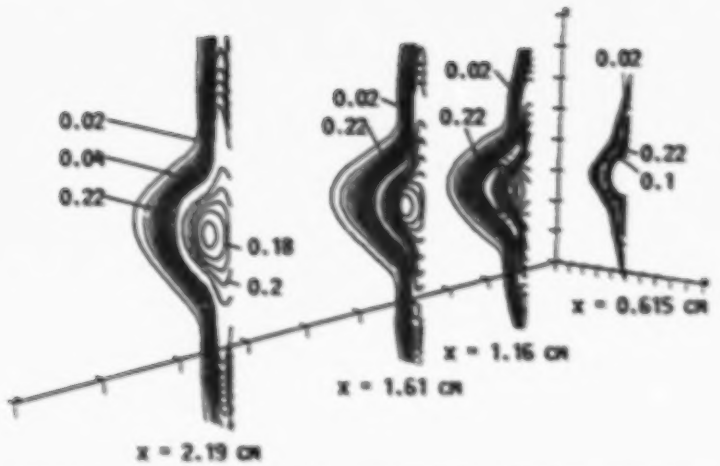
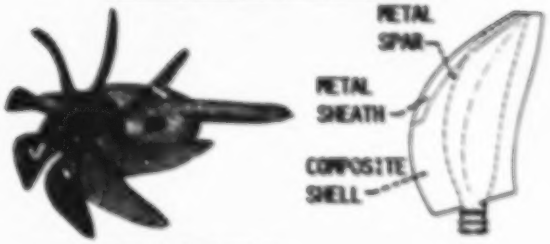
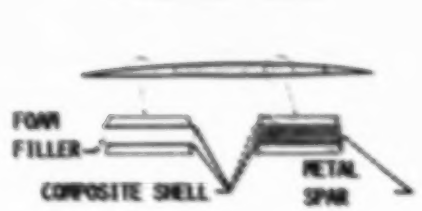



FIGURE 5. - RPLUS SIMULATION OF TRANSVERSE HYDROGEN INJECTION IN A SUPERSONIC AIRSTREAM - WATER VAPOR CONTOURS.




TURBOPROP STAGE AND PROPELLER




BLADE CONSTRUCTION

MULTIDISCIPLINARY ANALYSIS MODULES

- MATHEMATICAL OPTIMIZER
- BLADE MODEL GENERATOR
- AERODYNAMIC ANALYZER
- ACOUSTIC ANALYZER
- STRESS AND FREQUENCY ANALYZER
- AEROELASTIC FLUTTER ANALYZER
- FORCED VIBRATION ANALYZER

TYPICAL OPTIMIZATION RESULTS

| | INITIAL DESIGN | FINAL DESIGN |
|---|---|---|
| EFFICIENCY, PERCENT | 82.2 | 83.3 |
| NEAR-FIELD NOISE, dB | 144 | 137 |
| WEIGHT, LB | 41.1 | 41.2 |
| REDUCTION IN AIRCRAFT DIRECT OPERATING COST, PERCENT | | 3.32 |

FIGURE 6. - STRUCTURAL TAILORING OF ADVANCED TURBOPROPS. OPTIMUM PROPFAN DESIGN FOR MINIMUM OPERATING COST.

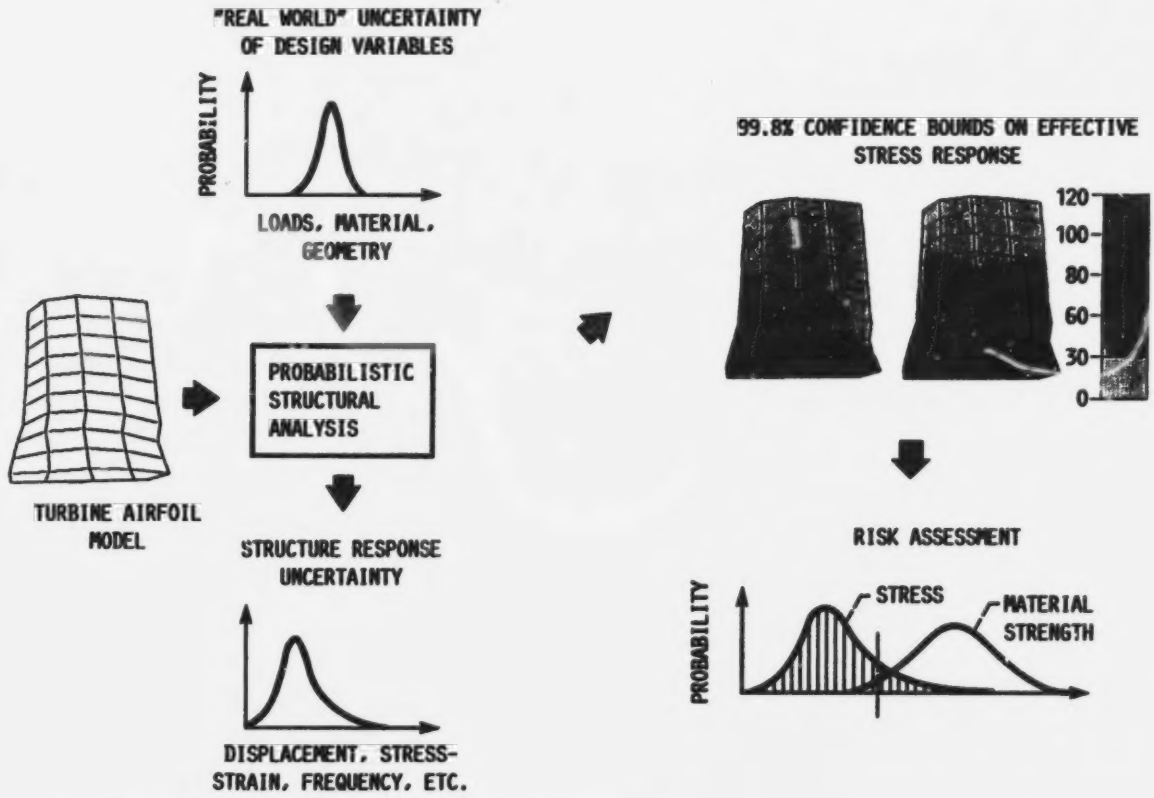
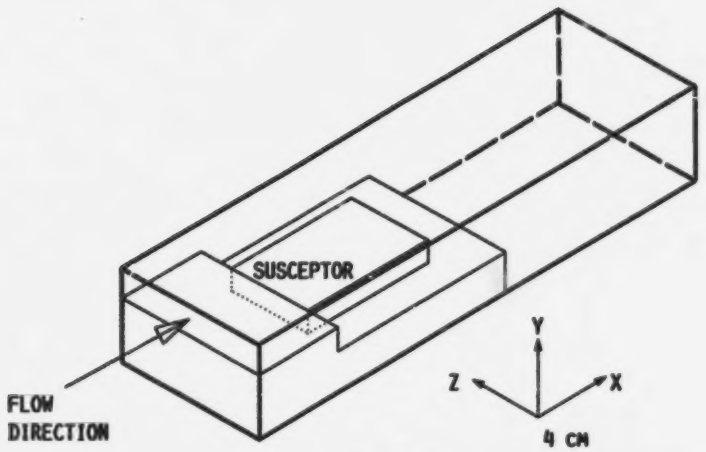
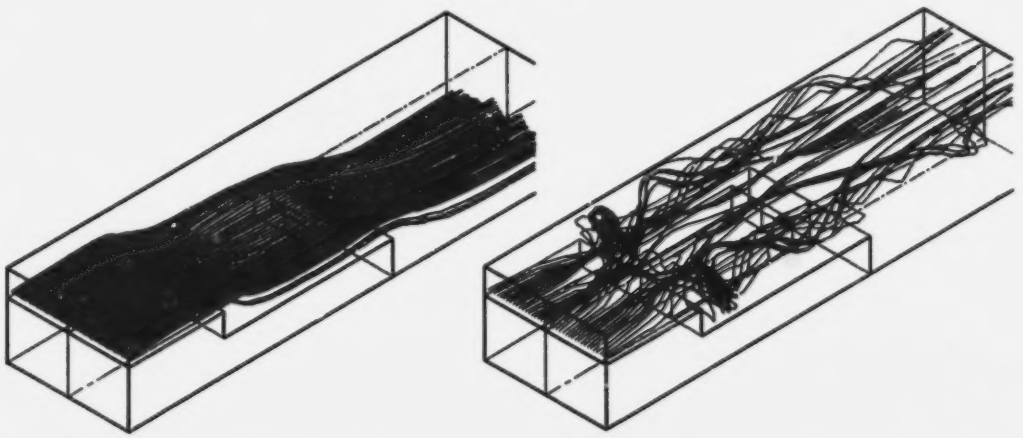



FIGURE 7. - PROBABILISTIC ANALYSIS AND DESIGN METHODOLOGY FOR ENGINE STRUCTURES RELIABLILITY AND RISK ASSESSMENT.




(A) REACTOR MODEL.



(B) PARTICLE TRAJECTORIES FOR $H_2$-Ar CARRIER AT 0 g.

(C) PARTICLE TRAJECTORIES FOR $H_2$-Ar CARRIER AT 1 g.

FIGURE 8. - 3-D ANALYSIS OF CHEMICAL VAPOR DEPOSITION.

ORIGINAL PAGE IS
OF POOR QUALITY

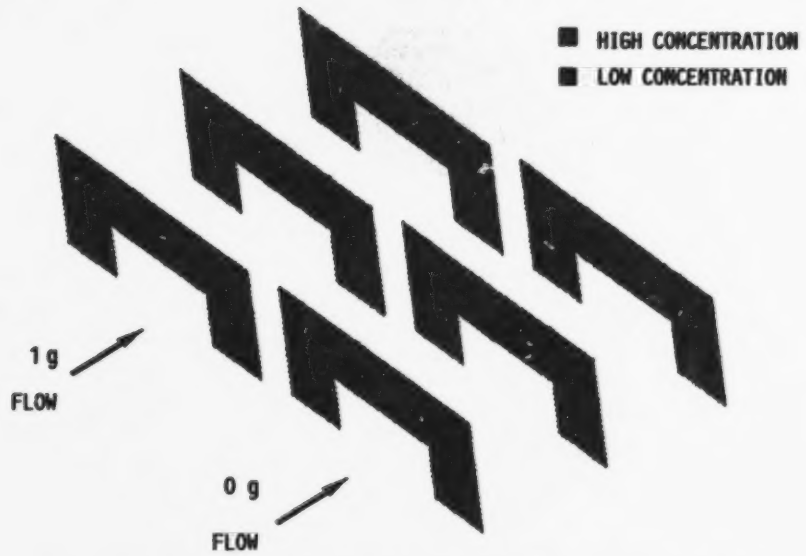



FIGURE 9. - PREDICTED SILANE CONCENTRATION FOR $H_2$-Ar CARRIER GAS.

FIGURE 10. - PARALLEL CALCULATIONS OF SSME FUEL TURBINE TEMPERATURE PROFILES USING CRAY X-MP.

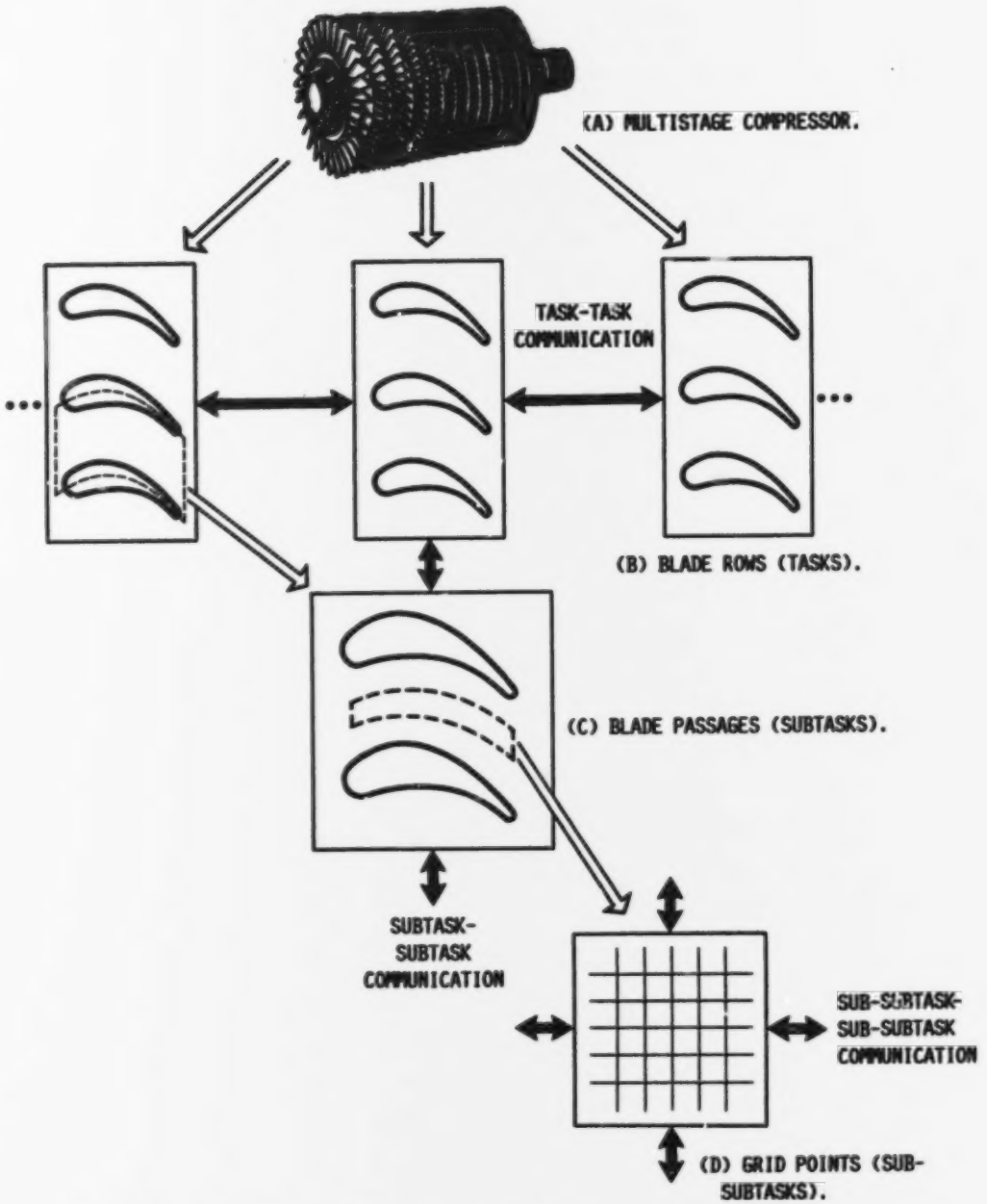



FIGURE 11. - HIERARCHY OF PARALLELISM IN MULTISTAGE TURBOMACHINERY CALCULATIONS.

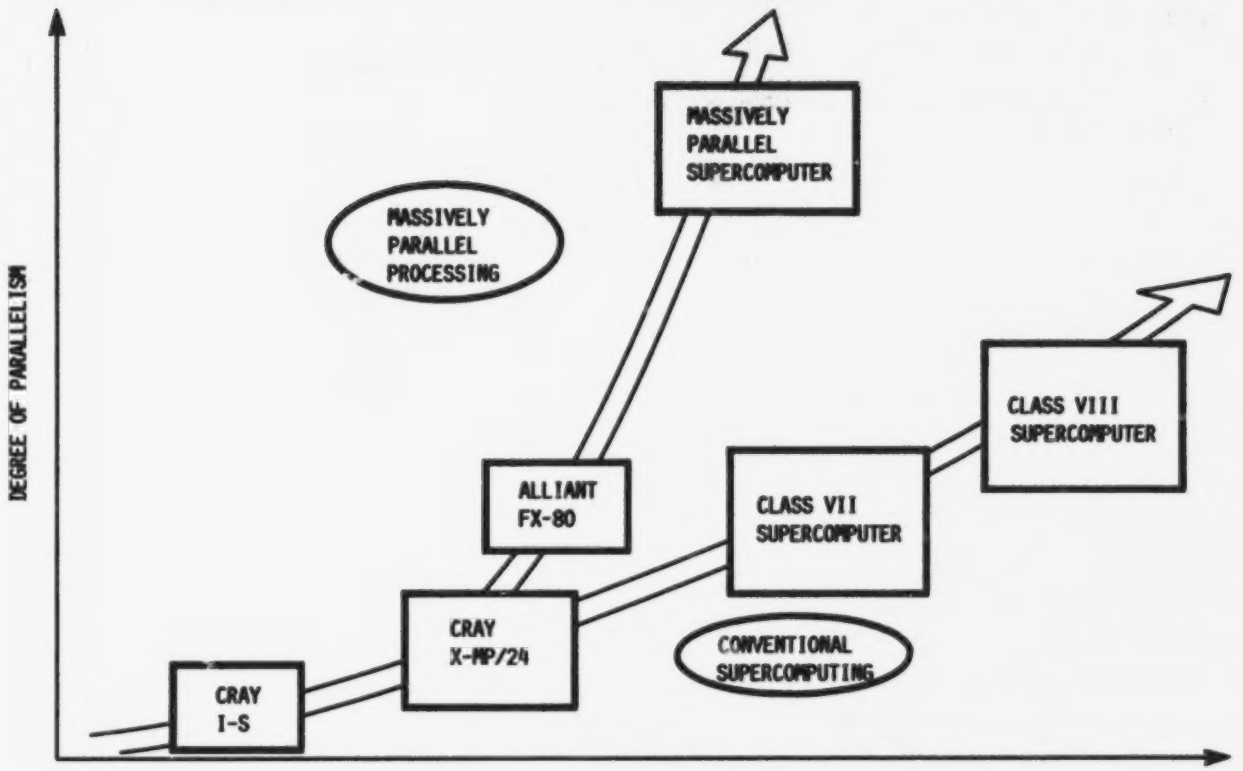



FIGURE 12. - LEWIS IMPLEMENTATION OF HIGH PERFORMANCE COMPUTERS.

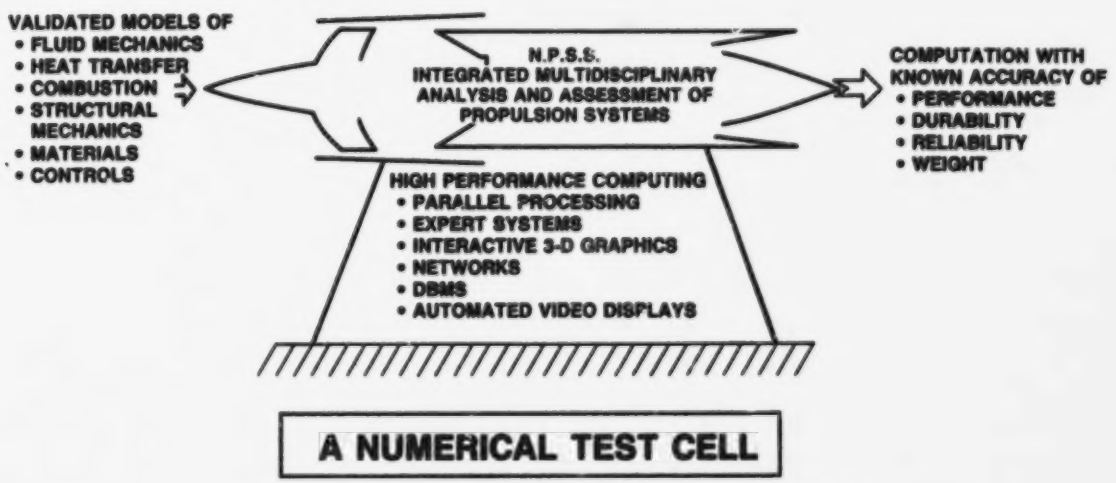



FIGURE 13. - NUMERICAL PROPULSION SYSTEM SIMULATION (NPSS).

NASA
National Aeronautics and Space Administration

# Report Documentation Page

| 1. Report No. | 2. Government Accession No. | 3. Recipient's Catalog No. |
|---|---|---|
| NASA TM-101987 | | |

**4. Title and Subtitle**
Advances in Computational Design and Analysis of Airbreathing Propulsion Systems

**5. Report Date**

**6. Performing Organization Code**

**7. Author(s)**
John M. Klineberg

**8. Performing Organization Report No.**
E-4689

**10. Work Unit No.**
505-62-21

**9. Performing Organization Name and Address**
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135-3191

**11. Contract or Grant No.**

**13. Type of Report and Period Covered**
Technical Memorandum

**12. Sponsoring Agency Name and Address**
National Aeronautics and Space Administration
Washington, D.C. 20546-0001

**14. Sponsoring Agency Code**

**15. Supplementary Notes**
Prepared for the Ninth International Symposium on Airbreathing Engines sponsored by the International Society for Air Breathing Engines, Athens, Greece, September 4-9, 1989.

**16. Abstract**

The development of ~~new~~ commercial and military aircraft depends, to a large extent, on engine manufacturers being able to achieve significant increases in propulsion capability through improved component aerodynamics, materials, and structures. The recent history of propulsion has been marked by efforts to develop computational techniques that can speed up the propulsion design process and produce superior designs. The availability of powerful supercomputers, such as the NASA Numerical Aerodynamic Simulator, and the potential for even higher performance offered by ~~new~~ parallel computer architectures, have opened the door to the use of multi-dimensional simulations to study complex physical phenomena in propulsion systems that have previously defied analysis or experimental observation. ~~This paper provides~~ an overview of several NASA Lewis research efforts is provided that are contributing toward the long-range goal of a "numerical test-cell" for the integrated, multidisciplinary design, analysis, and optimization of propulsion systems. Specific examples in Internal Computational Fluid Mechanics, Computational Structural Mechanics, Computational Materials Science, and High Performance Computing are cited and described in terms of current capabilities, technical challenges, and future research directions.

| 17. Key Words (Suggested by Author(s)) | 18. Distribution Statement |
|---|---|
| Computers; Propulsion design; Simulation; Fluid mechanics; Structural mechanics; Materials science | Unclassified - Unlimited<br>Subject Category 07 |

| 19. Security Classif. (of this report) | 20. Security Classif. (of this page) | 21. No of pages | 22. Price* |
|---|---|---|---|
| Unclassified | Unclassified | 18 | A03 |

NASA FORM 1626 OCT 86

*For sale by the National Technical Information Service, Springfield, Virginia 22161